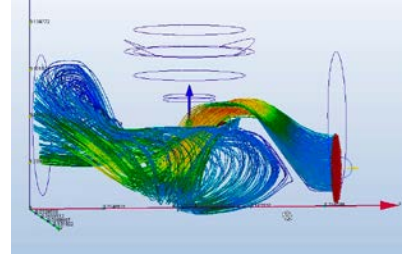


Lesson: Steady flow CFD analysis

In this lesson, you will conduct an analysis of fluid moving in a steady state and review the results of the analysis.

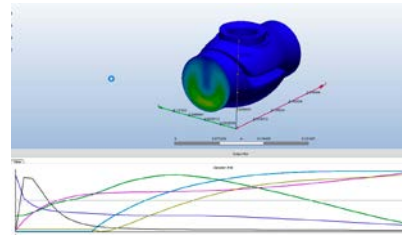
Learning Objectives:

- Set the criteria for a steady flow study.
- Run a steady flow study.
- Locate different methods to review the results.

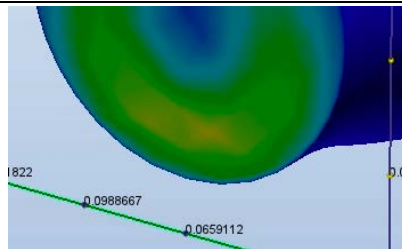


The completed exercise

1. Open the file Steady State.cfdst and then click Setup>Solve. Examine the options available in the Solve control panel, make no changes, then click Solve. Investigate the model.



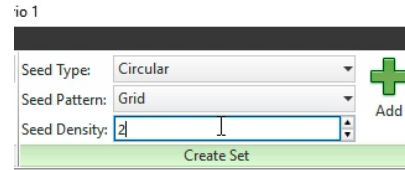
2. Note that the maximum velocity from the outlet is roughly 3.5 m/s. The input side begins at 1.5 m/s.



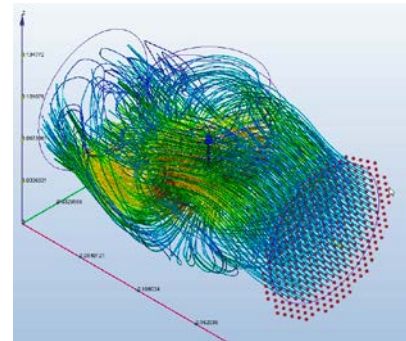
3. Click Results>Planes then Click Results>Add and a plane will appear. Align this to the Y plane. Selecting the arrows will allow you to move the plane.



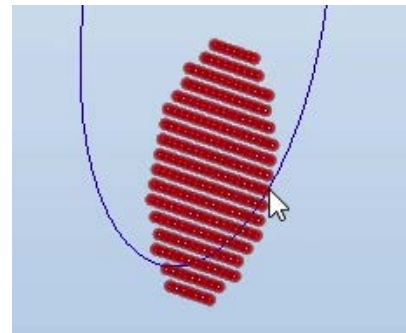
4. Click Results>Traces. In the Create Set section, change the Seed Type to Circular, the Seed Pattern to Grid, and the Seed Density to 2.



5. Click Setup>Add then click in the center of the inlet circle and drag outwards until the red dots engulf the circle; this will create the seeds. Examine the results. Note the areas of high velocity and low velocity as well as the areas where there is turbulence.



6. In the Browser, hide the current set of traces. Go back to the Create Set panel, change the Seed Density to 1, then click Add. When drawing the seeds, create a smaller patch. Use this to examine how the water from the lower part of the inlet acts.



7. Repeat the steps to examine how water from the upper part of the inlet acts. Also investigate Iso Surfaces and Iso Volumes and note how they help understand the flow through the design.

